TWO DIMENSIONAL ANALYSIS OF FRAME STRUCTURES UNDER ARBITRARY LOADING

A THESIS SUBMITTED IN

PARTIAL FULFILLMENT OF THE REQUIREMENT FOR

THE DEGREE OF

BACHELOR OF TECHNOLOGY

IN

CIVIL ENGINEERING

By

SHARBANEE PRUSTY (107CE026)

NIRAJ KUMAR AGRAWAL(107CE030)

Under the supervision of

Prof. K.C. Biswal

Department of Civil Engineering

National Institute of Technology, Rourkela

National Institute of Technology

Rourkela

CERTIFICATE

This is to certify that the thesis entitled, "**Two Dimensional Analysis Of Frame Structures under arbitrary loading**" submitted by **Miss Sharbanee Prusty** and **Mr. Niraj Kumar Agrawal** in partial fulfilment of the requirements for the award of Bachelor of Technology Degree in Civil Engineering at the National Institute of Technology, Rourkela is an authentic work carried out by them under my supervision and guidance.

To the best of my knowledge, the matter embodied in the thesis has not been submitted to any other University/Institute for the award of any Degree or Diploma.

Date:

Prof. Kishore Chandra Biswal

Department of Civil Engineering

National Institute of Technology

Rourkela- 769008

ACKNOWLEDGEMENT

We take this opportunity as a privilege to thank all individuals without whose support and guidance we could not have completed our project in this stipulated period of time. First and foremost we would like to express our deepest gratitude to our Project Supervisor Prof. K.C.Biswal, Department of Civil Engineering, National Institute of Technology, Rourkela for his invaluable support, motivation and encouragement throughout the period this work was carried out. His readiness for consultation at all times, his educative comments and inputs, his concern and assistance even with practical things have been extremely helpful.

Our deep sense of gratitude to Ms.Itishree Mishra, M.Tech (Res), Department of Civil Engineering, National Institute of Technology, Rourkela, for her support and guidance. She was there to help us throughout our experiments and helped us understand the technical aspects of the experiments.

We also extend our heartfelt thanks to our families, friends and the Almighty.

Sharbanee Prusty (107CE026)

Department of Civil Engineering

National Institute of Technology, Rourkela

Niraj Kumar Agrawal (107CE030)

Department of Civil Engineering

National Institute of Technology, Rourkela

CONTENTS

		Page No.
	Certificate	i
	Acknowledgement	ii
	List of figures	vi
	List of tables	viii
	Abstract	ix
1.	**INTRODUCTION**	**1**
	1.1.Literature Review	1
2.	**FINITE ELEMENT METHOD AND PLANE FRAMES**	**3**
	2.1. Numerical Analysis	3
	2.2.Plane Frame Element	3
	2.2.1.Frame Stiffness Matrix	4
	2.2.2.Formulation Of System Of Equations	5
3.	**DYNAMIC ANALYSIS OF FRAMES**	**7**
	3.1.Mass Matrix	7
	3.1.1.Construction of Mass Matrix	8
	3.1.2.Mass Matrix Properties	8
	3.2.Eigen Values and Eigen Vectors	9
	3.3.Mode Shapes	9
	3.4.Free Vibration: Damped and Undamped Systems	10
	3.5.Dynamic Analysis by Numerical Integration	11

3.5.1. Solution To Equation By Numerical Integration 12

3.5.1.1. Newmark Integration Method 12

4. ‘n’ BAY ‘n’ STOREY PLANE FRAME ANALYSIS 13

4.1. Steps Involved In 2-D Frame Analysis 14

5. EXPERIMENTAL ANALYSIS: FFT ANALYSER 16

6. RESULTS 18

6.1. Static Analysis 18

6.1.1. Comparison Of Results Obtained By Changing The Number Of Nodes 18

6.1.2. Study Of Deflection For Different Storey Frames With Single Bay 24

6.1.3.Variation of Base Moments With Different Number Of Storeys With Single Bay 26

6.1.4.Variation of Base Moments With Different Number of Bays and Fixed Number of Storeys 27

6.2.Dynamic Analysis 29

6.2.1 Comparison Of Experimental And Numerical Analysis Results 29

6.2.2. Study Of Variation Of Natural Frequency With Changing Number Of Storeys And Fixed Number Of Bay **31**

6.2.3. Study Of Variation Of Natural Frequency With Changing Number Of Bays And Fixed Number Of Storeys **33**

6.2.4. Dynamics Analysis Under Different Load Conditions **35**

6.2.4.1. Systems Without Damping **35**

6.2.4.2. Systems With Damping **36**

7. CONCLUSION **43**

8. REFERENCES **44**

LIST OF FIGURES

		Page no.
1.1	Plane Frame Element	4
4.1	Multi Storey, Multi bay frame	13
5.1	FFT Analyser	17
5.2	Impact Hammer	17
6.1	One storey, one bay frame with zero intermediate nodes in beams and columns	19
6.2	Shear Force Diagram	20
6.3	Shear Force Diagram	20
6.4	Axial Force Diagram	20
6.5	One storey, one bay frame with one intermediate node in beams and columns	21
6.6	Shear Force Diagram	22
6.7	Bending Moment Diagram	22
6.8	Axial Force Diagram	22
6.9	Multi-storey, single bay frame	24
6.10	Variation of deflection in X-axis	25
6.11	Variation of deflection in Y-axis	25
6.12	Variation of base moments with number of storeys	26
6.13	Multi-bay frame with 10 storeys	27
6.14	Variation of base moments with number of bays	28
6.15	Multi-storey, single bay frame	31
6.16	Variation of modal frequency with number of storeys considering 1-60 storeys	32
6.17	Variation of modal frequency with number of storeys considering 10-60 storeys	33
6.18(a)	Variation of Modal Frequency with changing number of bays: Mode 1	33

6.18(b)	Variation of Modal Frequency with changing number of bays: Mode 2	34
6.18(c)	Variation of Modal Frequency with changing number of bays: Mode 3	34
6.19	Elcentro force data	35
6.20	Mode diagram for the system	36
6.21	Forced Vibration Deflections Due To Arbitrary Load on Ground (Elcentro force data): Undamped System	36
6.22	IS Code Force Data	37
6.23	Forced Vibration Deflection Due To Arbitrary Load On Ground (IS Code): Undamped System	37
6.24	Forced Vibration Deflection Due To Load on Frame: Undamped System	38
6.25	Forced Vibration Deflection Due To Sinusoidal Load on Ground: Undamped System	39
6.26	Forced Vibration Deflections Due To Arbitrary Load on Ground (Elcentro force data):Damped System	40
6.27	Forced Vibration Deflection Due To Arbitrary Load On Ground (IS Code): Damped System	40
6.28	Forced Vibration Deflection Due To Load on Frame :Damped System	41
6.29	Forced Vibration Deflections Due To Sinusoidal Load on Ground: Damped System	41

LIST OF TABLES

Page No.

6.1	Assumed date for the frame	18
6.2	Displacements Obtained for zero intermediate nodes in beams and columns	19
6.3	Forces obtained for zero intermediate nodes in beams and columns	19
6.4	Displacements Obtained for one intermediate node in beam and column	21
6.5	Forces obtained for one intermediate node in beam and column	21
6.6	Deflection results for varying number of storeys	24
6.7	Variation of base moments with number of storeys	26
6.8	Variation of base moments with number of bays	27
6.9	Details of the experimental model	29
6.10	Comparison of numerical and experimental analysis results	30
6.11	Variation of Modal Frequencies with number of storeys	31
6.12	Variation of Modal Frequencies with number of bays	33
6.13	Assumed data for the system	35
6.14	Comparison of results in undamped and damped conditions	36

ABSTRACT

In this modern era all the high rise buildings are designed as multi story and multi-bay frame structures. When such structures are subjected to various loads or displacements they behave both statically and dynamically. When the loads act slowly the inertia forces are neglected so only static load analysis is justified but when the load act very fast dynamic analysis is also considered.

This project deals with two dimensional analysis of plane frame under arbitrary ground motions and load conditions. Finite Element method has been used for numerical analysis. Static and dynamic response of the plane frame element was obtained by writing a code in matlab. A study of variation static properties and dynamic properties with different numbers of storeys and bays in the frame element has been done. The effect of dampers in reducing the displacement under forced vibration has been studied. The numerical analysis has been experimentally verified by determination of natural frequency of plane frame model using FFT analyzer.

1. INTRODUCTION

Reference to recent and current technical literature indicates that the problem of dynamics of building frames are topics of great importance chiefly because there is need of practicable and accurate techniques to be used in design of such structures.

The accurate determination of deflection, stresses and vibrational movements is necessary in such cases. In engineering design, the real behaviour of a structure is provided by determining geometrical, damping and mass and connection model well. [1] Thus is the importance of dynamic and static analysis of framed structures under various conditions. The concept of finite element method is used for numerical analysis where in the model at hand is sub divided into components called finite elements and response of each element is expressed in terms of a finite number of degrees of freedom characterized as the value of an unknown function. [2] We have studied the various properties of a plane frame element and developed a MATLAB code for two dimensional numerical analysis of a plane frame. The analyses made have been done on the assumption that the joint connections are fully rigid.

1.1.LITERATURE REVIEW

Ali Ugur Ozturk and Hikmet H. Catal, in their paper "Dynamic Analysis of semi-rigid frames" examined semi-rigid frames having same geometry and cross-section but different spring coefficients. The semi-rigid frame was modelled by rotational springs and stiffness matrix was obtained using rigidity index at the ends of a semi-rigid frame element. In a semi-rigid frame, an increase in rate between length of bay and height of storey (L/h) causes reduction in coefficient and lateral rigidity decreases. The study indicates that the connection flexibility tends to increase periods, especially in lower modes, while it tends to decrease frequency. [1]

Miodrag Sekulonic *et al.* studied the effects of flexibility and damping in nodal connections in the paper " Dynamic analysis of steel frames with semi-rigid connections". A numerical model that includes both non-linear connection behaviour and geometric non-linearity of the structures had been developed. An increase in connection flexibility reduces the frame stiffness and thus the eigen functions. [3].

Shousuke Moring and Yasuhiro Uchida in their paper "Dynamic response of steel frames under earthquake excitation in horizontal arbitrary direction" performed a dynamic analysis of the elasto-plastic response of one-storey single bay space frames. It is shown that displacement responses under the two –directional excitation are larger than those under the one-directional excitation, which lead to an earlier collapse in the earlier case.[4]

The thesis has been divided into various chapters for a better representation.

Chapter 2 discusses the various attributes of the numerical analysis, matrices involved in the calculations for static analysis, formulation of the stiffness matrix and the system of equations, which are subsequently used in the code for obtaining the results.

Chapter 3 focuses on dynamic analysis, properties of mass matrix and its formulation, significance of eigen values and eigen vectors and construction of mode shapes. The equations involved in the study of damped and undamped conditions of a system are also discussed.

Chapter 4 gives an idea about the basic algorithm followed in formulation of the code

Chapter 5 explains the experimental set up used for obtaining experimental results.

Chapter 6 dwells upon the analysis done by us as a part of the project and representation of various outputs obtained in tables and graphs and drawing inferences.

2.1. NUMERICAL ANALYSIS

Finite Element Method is a numerical procedure for solving engineering problems. We have assumed linear elastic behaviour throughout. The various steps of finite element analysis are:

1) Discretizing the domain wherein each step involves subdivision of the domain into elements and nodes. While for discrete systems like trusses and frames where the system is already discretized upto some extent we obtain exact solutions, for continuous systems like plates and shells , approximate solutions are obtained
2) Writing the element stiffness matrices: The element stiffness equations need to be written for each element in the domain. For this we have used MATLAB.
3) Assembling the global stiffness matrix: We have used the direct stiffness approach for this.
4) Applying boundary conditions: The forces, displacements and type of support conditions etc. are specified
5) Solving the equations: The global stiffness matrix is partitioned and resulting equations are solved.
6) Post processing: This is done to obtain additional information like reactions and element forces and displacements.

2.2. PLANE FRAME ELEMENT

A plane frame element is a two-dimensional finite element. It is expressed in both local and global coordinates. In the case of plane frame, all the members lie in the same plane and are interconnected by rigid joints. The internal stress resultants at a cross-section of a plane frame member consist of bending moment, shear force and an axial force. The significant deformations in the plane frame are only flexural and axial. Initially, the stiffness matrix of the plane frame member is derived in its local co-ordinate axes and then it is transformed to global co-ordinate system. In the case of plane frames, members are oriented in different directions and hence before forming the global stiffness matrix it is necessary to refer all the member stiffness matrices to the same set of axes. This is achieved by transformation of forces and displacements to global co-ordinate system.[11]

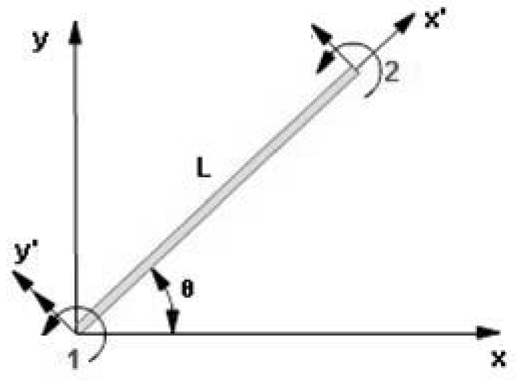

Fig.1.1 Plane Frame Element

2.2.1. FRAME STIFFNESS MATRIX

The plane frame element has modulus of elasticity E, moment of inertia I, cross-sectional area A, and length L. Each plane frame element has two nodes and is inclined with an angle θ measured counter clockwise from the positive global X axis. A plane frame element has six degrees of freedom: three at each node (two displacements and a rotation). Sign convention used is that displacements are positive if they point upwards and rotations are positive if they are counter clockwise. [5]

Let C= cosθ

S= sinθ

k = Element stiffness matrix

Then the element stiffness matrix k is given as :

$$k = \frac{E}{L} \times$$

$$\begin{bmatrix} AC^2 + \frac{12L}{L^2} & \left(A - \frac{12L}{L^2}\right)CS & -\frac{6I}{L}S & -\left(AC^2 + \frac{12I}{L^2}S^2\right) & -\left(A - \frac{12I}{L^2}\right)CS & -\frac{6I}{L}S \\ & AS^2 + 12\frac{12I}{L^2}C^2 & \frac{6I}{L}C & -\left(A - \frac{12I}{L^2}\right)CS & -\left(AS^2 + \frac{12I}{L^2}C^2\right) & \frac{6I}{L}C \\ & & 4I & \frac{6I}{L}S & -\frac{6I}{L}C & 2I \\ & & & AC^2 + \frac{12L}{L^2}S^2 & \left(A - \frac{12I}{L^2}\right)CS & \frac{6I}{L}S \\ & & & & AS^2 + 12\frac{12I}{L^2}C^2 & -\frac{6I}{L}C \\ \text{Symmetry} & & & & & 4I \end{bmatrix}$$

2.2.2. FORMULATION OF SYSTEM OF EQUATIONS

For a structure with n nodes, the global stiffness matrix K will be of size 3nX3n.

After obtaining K, we have:

[K]{U}= {F}

Where U is the global nodal displacement vector and F is the global nodal force vector.

At this step the boundary conditions are imposed manually to vectors U and F to solve this equation and determine the displacements. By post processing, the stresses, strains and nodal forces can be obtained.

{f} = [k’] [R]{u},

Where {f} is the 6 X1 nodal force vector in the element and u is the 6X1 element displacement vector.

The matrices [k’] and [R] are given by the following:

$$[k']=\begin{bmatrix} \frac{EA}{L} & 0 & 0 & -\frac{EA}{L} & 0 & 0 \\ 0 & \frac{12EI}{L3} & \frac{6EI}{L2} & 0 & -\frac{12EI}{L3} & \frac{6\mathrm{EI}}{\mathrm{L2}} \\ 0 & \frac{6EI}{L2} & \frac{4EI}{L} & 0 & \frac{6EI}{L2} & \frac{2EI}{L} \\ -\frac{EA}{L} & 0 & 0 & \frac{EA}{L} & 0 & 0 \\ 0 & -\frac{12EI}{L3} & -\frac{6EI}{L2} & 0 & -\frac{12EI}{L3} & -\frac{6EI}{L2} \\ 0 & \frac{6EI}{L2} & \frac{2EI}{L} & 0 & -\frac{6EI}{L2} & \frac{4EI}{L} \end{bmatrix}$$

$$[R]=\begin{bmatrix} C & S & 0 & 0 & 0 & 0 \\ -S & C & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & C & S & 0 \\ 0 & 0 & 0 & -S & C & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \end{bmatrix}$$

Where C=cosθ; S=sinθ

The first and second elements in each vector {u} are the two displacements while the third element is the rotation, respectively, at the first node, while the fourth and fifth elements in each vector are the two displacements while the sixth element is the rotation, respectively ,at the second node.[5]

3. DYNAMIC ANALYSIS OF FRAMES

Static analysis holds when the loads are slowly applied. When the loads are suddenly applied or when the loads are of variable nature, effects of mass and acceleration come into picture. If a solid body is deformed elastically and suddenly released, it tends to vibrate about its equilibrium position. This periodic motion due to the restoring strain energy is called free vibration. The number of cycles per unit time is called frequency and the maximum displacement from the equilibrium position is the amplitude.

The dynamic analysis of plane frame elements includes axial effects in the stiffness and mass matrices. It also requires a coordinate transformation of the nodal coordinates from element or local coordinates to system or global coordinates, so that appropriate superposition can be applied to assemble the system matrices.

We study the required matrices for consideration of axial effects as well as matrix required for the transformation of coordinates. A computer program in the form of MATLAB code is developed for both static and dynamic analysis of plane frames.

3.1. MASS MATRIX

The construction of the master mass matrix **M** largely parallels that of the master stiffness matrix K. Mass matrices for individual elements are formed in local coordinates, transformed to global, and merged into the master mass matrix following exactly the same techniques used for K. In practical terms, the assemblers for K and M can be made identical.

$$\begin{Bmatrix} P_1 \\ P_2 \\ P_3 \\ P_4 \\ P_5 \\ P_6 \end{Bmatrix} = \begin{bmatrix} 140 & 0 & 0 & 0 & 0 & 0 \\ 0 & 156 & 0 & 0 & 0 & 0 \\ 0 & 22L & 4L^2 & 0 & 0 & 0 \\ 70 & 0 & 0 & 140 & 0 & 0 \\ 0 & 54 & 13L & 0 & 156 & 0 \\ 0 & -13L & -3L^2 & 0 & -22L & 4L^2 \end{bmatrix} \begin{Bmatrix} \delta_1 \\ \delta_2 \\ \delta_3 \\ \delta_4 \\ \delta_5 \\ \delta_6 \end{Bmatrix}$$

3.1.1. CONSTRUCTION OF MASS MATRIX

The master mass matrix is built up from element contributions, and we start at that level. The construction of the mass matrix of individual elements can be carried out through several methods. These can be categorized into three groups: direct mass lumping, variational mass lumping, and template mass lumping. In direct mass lumping, the total mass of element e is directly apportioned to nodal freedoms and a diagonally lumped mass matrix is formed. A key motivation for direct lumping is that a diagonal mass matrix may offer computational and storage advantages in certain simulations, notably explicit time integration.

In variational mass lumping, a second class of mass matrix construction methods are based on a variational formulation. This is done by taking the kinetic energy as part of the governing functional.

3.1.2.MASS MATRIX PROPERTIES

Mass matrices must satisfy certain conditions that can be used for verification and debugging which are as follows.

- **Matrix Symmetry**: This means $(M^e)^T = M$, which is easy to check.
- **Physical Symmetries**: Element symmetries must be reflected in the mass matrix.
- **Conservation:** At a minimum, total element mass must be preserved which can be checked by applying a uniform translational velocity and checking that linear momentum is conserved. Higher order conditions, such as conservation of angular momentum, are optional and not always desirable.
- **Positivity:** This constraint is non linear in the mass matrix entries. It can be checked in two ways: through the Eigen values of M^e or the sequence of principal minors. The second technique is more practical if the entries of M^e are symbolic.

3.2. EIGEN VALUES AND EIGEN VECTORS

The generalised problem in free vibration is that of evaluating an Eigen value λ, which is measure of the frequency of vibration together with the corresponding Eigen vector **U** indicating the mode shape as in $\mathbf{KU = \lambda M}$

Properties of Eigen Vectors:

For a positive definite symmetric stiffness matrix of size *n,* there are *n* real Eigen values and corresponding Eigen vectors satisfying the above equation.

$$0 \leq \lambda 1 \leq \lambda 2 \leq \ldots\ldots \leq \lambda_n$$

If U_1, U_2....U_n are the corresponding Eigen vectors, we have

$$\mathbf{KU_i = \lambda_i MU_i}$$

The Eigen vectors possess the property of being orthogonal with respect to both the stiffness and mass matrices. The lengths of Eigen vectors are generally normalized so that

$$\mathbf{U_i^T\, MU_i = 1}$$

The foregoing normalization of the Eigen vectors leads to the relation

$$\mathbf{U_i^T\, KU_i} = \lambda_i$$

The length of an Eigen vector may be fixed by setting its largest component to a preset value, say, unity.

3.3. MODE SHAPES

A mode shape is a particular pattern of vibration carried by a system at a particular frequency. For different frequencies, there are different mode shapes which are associated with it. The experimental technique of modal analysis discovers these mode shapes and the frequencies. The eigenvectors define the displacement configurations of the various modes of the system. Each mode has a natural frequency associated with it i.e. the Eigen value. For an mdof (multiple degrees of freedom) system with N degrees of freedom, N mode shapes and N frequencies will exist. The primary mathematical advantage of determining mode shapes is that they will be orthogonal to each other. For design engineers, mode shapes are useful because they represent the shape that the building will vibrate in free motion. These same shapes tend to dominate the motion during an earthquake (or windstorm). By understanding the modes of vibration, we can better design the building to withstand earthquakes.

Generally, the first mode of vibration is the one of primary interest. The first mode usually has the largest contribution to the structure's motion. The period of this mode is the longest. The first Eigen vector represents the shortest natural frequency. Hence, natural frequency is higher for subsequent higher eigenvectors. The higher order modes can be distinguished by the number of vibrational nodes. These are the points where the mode shape displacement remains zero (no lateral movement). The 2D mode shape will equal the number of vibrational nodes.[8]

3.4. FREE VIBRATION: DAMPED AND UNDAMPED SYSTEMS

A structure is said to be undergoing free vibration when it is disturbed from its static equilibrium state and then allowed to vibrate without any external dynamic excitation. Damping ratio regulates the rate of decay of motion in free vibration.

A system subjected to dynamic excitations is governed by the equation-

$$\mathbf{m\ddot{u} + c\mathring{u} + ku = p(t)}$$

For free vibration, $\mathbf{p}(t) = \mathbf{0}$; $\mathbf{m\ddot{u} + c\mathring{u} + ku = 0}$

For **systems without damping**, the equation governing the system is $\mathbf{m\ddot{u} + ku = 0}$ A solution for u(t) is found that satisfies the initial conditions of $\mathbf{u=u(0)}$ **and** $\mathbf{\mathring{u}=\mathring{u}(0)}$ at **t=0.**

The Eigen value problem is solved for natural frequencies and modes and a general solution to the above equation is given by superposition of the response in individual modes given by: $\mathbf{u}(t) = \Phi_n (A_n \cos \omega_n t + B_n \sin \omega_n t)$

For **systems with damping**, the free vibration response of the system is governed by

p(t)=0: **mü +ců+ ku = 0. [9]**

3.5. DYNAMIC ANALYSIS BY NUMERICAL INTEGRATION

Generally for getting the solution of the dynamic response of any given structural system we use the direct numerical integration of the dynamic equilibrium equations. Dynamic equilibrium is satisfied at discrete points in time, after the solution at time zero is defined. Time intervals of equal magnitude at Δt, 2Δt, 3Δt... NΔt are used. The integrations methods can be implicit or explicit. Explicit methods instead of involving the solution of a set of linear equations at each step, they use the differential equation at time "t" to predict a solution at time "t+Δt". For most real structures, which contain stiff elements, a very small time step is required in order to obtain a stable solution. Therefore, all explicit methods are conditionally stable with respect to the size of the time step.

Implicit methods attempt to satisfy the differential equation at time "t" after the solution at time "t-Δt" is found. These methods require the solution of a set of linear equations at each time step; however, larger time steps may be used. Implicit methods can be conditionally or unconditionally stable.[10]

3.5.1. SOLUTION TO EQUATION BY NUMERICAL INTEGRATION

The Numerical Solution can be calculated by various methods:

- Duhamel Integral
- Newmark Integration method
- Central difference Method
- Houbolt Method
- Wilson Method

We have followed the Newmark Integration Method in our MATLAB code.

3.5.1.1. Newmark Integration Method

The steps involved are:

I. **INITIAL CALCULATION**

A. Formulation of stiffness matrix K, mass matrix M and damping matrix C

B. Specification of integration parameters β and γ

C. Calculation of integration constants

$b_1 = 1/\beta\,\Delta t^2$; $b_2 = \frac{1}{\beta \Delta t}$; $b_3 = \beta - ½$; $b_4 = \gamma\,\Delta t\, b_1$;

$b_5 = 1+\gamma \Delta t b_2$; $b_6 = \Delta t\,(1+\gamma b_3 - \gamma)$

D. Formulation of effective stiffness matrix $\mathbf{K}^* = \mathbf{K} + b_1\mathbf{M} + b_4 C$

E. Triangulation of effective stiffness matrix $\mathbf{K}^* = \mathbf{LDL}^T$

F. Specification of initial conditions

II. FOR EACH TIME STEP t=Δt, 2Δt, 3Δt...... NΔt

A. Calculation of effective load vector

B. Solving for node displacement vector at time t

$L\,D\,L^T u_t = Ft^*$ (forward and back-substitution only)

C. Calculation of node velocities and accelerations at time **t**

D. Go to Step II.A with t=t+Δt. **[10]**

4. ‘n’ BAY ‘n’ STOREY PLANE FRAME ANALYSIS

This section deals with the dynamic and static analysis of a **‘n’ bay ‘n’ storey plane frame**. The code written in MATLAB determines the **nodal displacements, forces** and **end moments** at various nodes in beams and column element of plane frame as a part of static analysis. The code also determines the **free vibration frequency** and plots **mode shapes of various fundamental frequencies** as a part of dynamic analysis. Under various load conditions such as sinusoidal load on ground, arbitrary load on ground and load on frame, the forced vibration analysis is done and a graph between time and displacement is plotted at various natural frequencies.

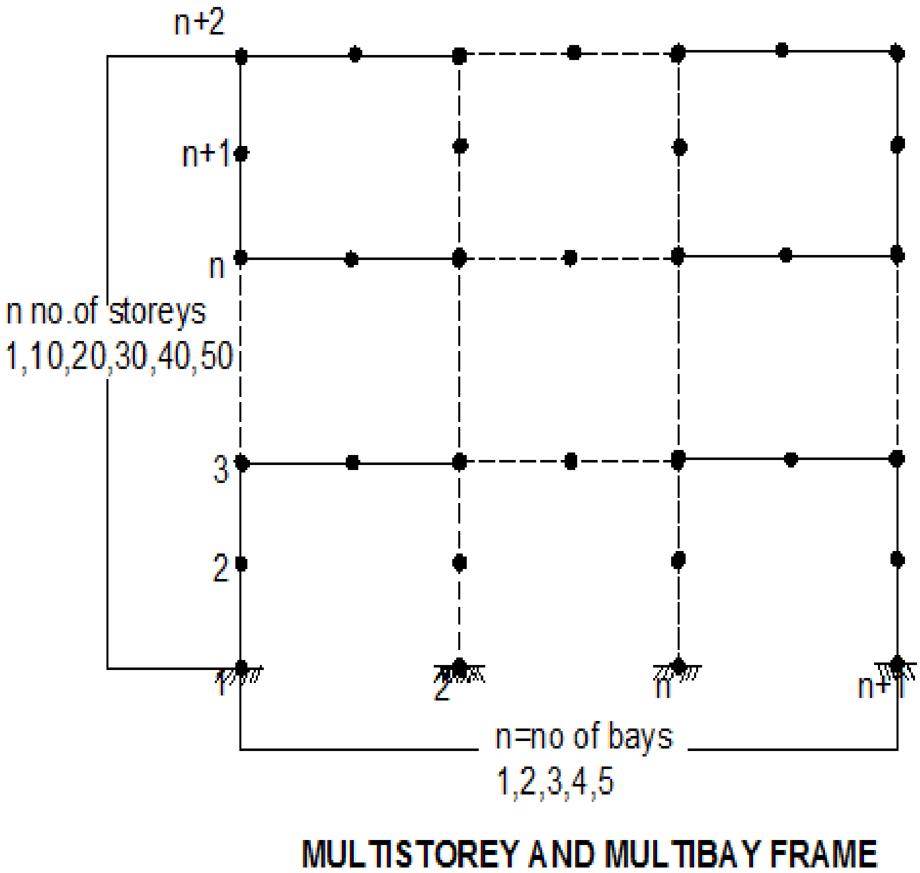

Fig 4.1.Multi Storey, Multi bay frame

4.1. STEPS INVOLVED IN 2-D FRAME ANALYSIS

A MATLAB program for **2D static and dynamic analysis of multi storey and multi bay frame with intermediate nodes in columns and beams** is formulated.

Enter the details of plane frame i.e. the number of intermediate nodes in columns and beams, number of storeys and bays, Elasticity modulus of material of frame, area, moment of inertia of columns and beams.

↓

Calculation of total no of nodes in each column, total no nodes and elements in frame.

↓

Generation of X and Y coordinate for each node.

↓

Nodal connectivity is done for each beam and column element i.e. to generate node pattern for the multi storey multi bay frame.

↓

Length and angle calculations for each element from generated coordinates.

↓

Calculation of local stiffness and mass matrix for each frame element using plane frame element stiffness and plane frame element mass function

↓

Assembling of local stiffness matrix into global stiffness and mass matrix using plane frame assemble 1 function

↓

Applying Boundary Condition to get appropriate mass and stiffness matrix

STATIC ANALYSIS starts...

Generation of Force matrix using user's input.

Obtaining the displacement matrix from the static formula for frame

[K] * [u] = [F].

Generation of Global stiffness matrix and Force matrix

Plotting axial force, shear force and bending moment at each node w.r.t to length by using functions like plane frame element shear force, axial force and bending moment diagram.

DYNAMIC ANALYSIS starts...

Calculation of free vibration frequency from formula $[K] - \omega 2[M] = 0$.

The Eigen values and Eigen vectors of each element are calculated using in built mat lab function *eig*.

Various modes shapes of different fundamental frequency are plotted

Forced vibration analysis is done for different load conditions such as arbitrary load on ground, load on frame, sinusoidal load on ground.

The forced analysis is done by using New mark's beta method where effective stiffness and effective reaction matrix is calculated

Displacement is calculated by formula $u=k_{eff}/r_{eff}$ and a graph is plotted between time and displacement to get a resonance peak.

5. EXPERIMENTAL ANALYSIS: FFT ANALYSER

The Fourier transform is a mathematical procedure that was invented by Jean-Baptiste-Joseph Fourier in the early 1800's. The Fourier Transform yields the frequency spectrum of a time domain function. It is defined for continuous (or analog) functions. The FFT computes a discretized (sampled) version of the frequency spectrum of sampled time signal known as Discrete Fourier Transform (DFT).

FFT is a linear, one-to-one transformation that uniquely transforms the vibration signal from a linear dynamic system into its correct digital spectrum, and vice versa. If a signal contains any additive Gaussian random noise or randomly excited non-linear behavior, these portions of the signal are transformed into spectral components that appear randomly in the spectrum.

FFT Analyzers can be classified into two categories:

- Single channel
- Multi-channel.

Each channel can process a unique signal. Single channel analyzers are the most popular because they cost less, but they also have limited measurement capability. The distinguishing feature of a multi-channel analyzer is that all channels are simultaneously sampled. It is also assumed that filtering and other signal conditioning match within acceptable tolerances among all channels. If an analyzer has multiple channels, but they are multiplexed instead of simultaneously sampled, then each channel must be treated like a single channel analyzer channel. Simultaneously sampled signals contain the correct magnitudes & phases relative to one another, since they are all sampled at the same moments in time.

We use PULSE software as a platform for vibration analysis.

Fig. 5.1 FFT Analyser

Fig.5.2 Impact Hammer

6. RESULTS

The static and dynamic response of a plane frame under varying load and boundary conditions have been obtained and tabulated and its implications have been studied.

6.1. STATIC ANALYSIS

This section deals with the static analysis of the plane frame and determining the nodal displacements and forces for a given lateral load by finite element programming using a MATLAB code.

6.1.1. COMPARISON OF RESULTS OBTAINED BY CHANGING THE NUMBER OF NODES

The program is run for different storey buildings and the values of deflection and forces at each node are obtained.

For the first problem, we consider a one-storey building and by changing the number of nodes in the columns and the beams, we compare the results obtained at each node.

Assumed data [5]:

Table 6.1.Assumed date for the frame

Modulus of Elasticity, E	210 GPa (Steel)
Moment of Inertia, I	$5 * 10^{(-5)}$ m^4
Area of cross-section, A	$2 * 10^{(-2)}$ m^2
Length of the column	3 m
Length of the beam	4 m

CASE 1:

Total number of intermediate nodes in each column= 0
Total number of intermediate nodes in each beam = 0
Number of storeys= 1
Total number of nodes in the plane frame=4
Total number of elements in the plane frame=3

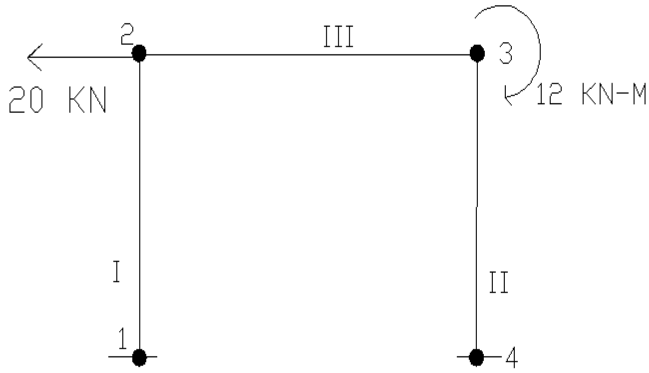

Fig. 6.1 One storey, one bay frame with zero intermediate nodes in beams and columns

Table 6.2 Displacements Obtained for zero intermediate nodes in beams and columns

Element number	Node 1	Node 2						
				Node 1			Node 2	
			x axis (Dx)	y axis (Dy)	rotation(Θ)	x axis (Dx)	y axis (Dy)	rotation(Θ)
I	1	2	0	0	0	-0.0038	0	0.0018
II	3	4	-0.0038	0	0.0014	0	0	0
III	2	3	-0.0038	0	0.0018	-0.0038	0	0.0014

Table 6.3. Forces obtained for zero intermediate nodes in beams and columns

Element No.	Node 1	Node 2			Forces(KN)			
				Node 1			Node 2	
			x axis(Fx)	y axis(Fy)	Moment	x axis(Fx)	y axis(Fy)	Moment
I	1	2	8.5865	-12.1897	-21.0253	-8.5865	12.1897	-15.5438
II	3	4	-8.5865	-7.8103	-6.8023	8.5865	7.8103	-16.6286
III	2	3	-7.8103	8.5865	15.5438	7.8103	-8.5865	18.8023

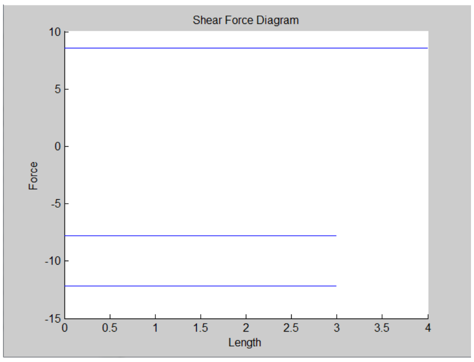

Fig. 6.2.Shear Force Diagram

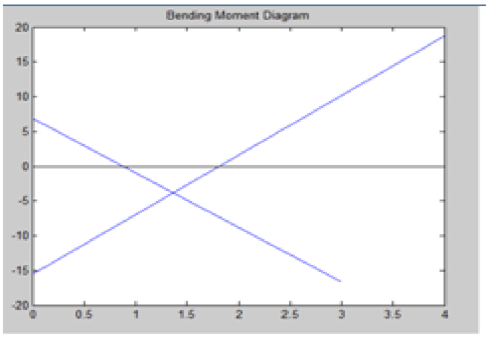

Fig.6.3 Bending Moment Diagram

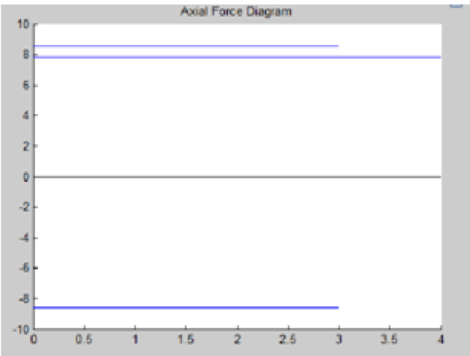

Fig. 6.4 Axial Force Diagram

CASE 2:

Total number of nodes in each column= 1
Total number of nodes in each beam = 1
Number of storeys= 1
Total number of nodes in the plane frame=7
Total number of elements in the plane frame= 6

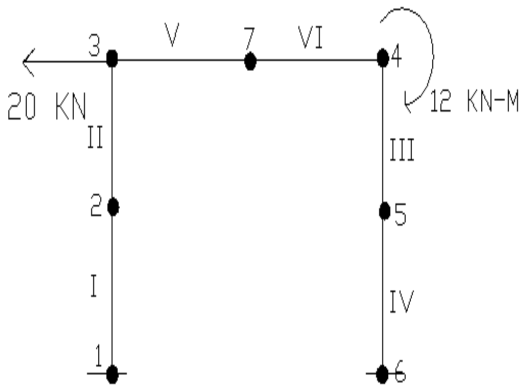

Fig. 6.5 One storey, one bay frame with one intermediate node in beams and columns

Table 6.4 Displacements Obtained for one intermediate node in beam and column

Element number	Node 1	Node 2						
				Node 1			Node 2	
			x axis(Dx)	y axis(Dy)	rotation(Θ)	x axis(Dx)	y axis(Dy)	rotation(Θ)
1	1	2	0	0	0	-0.0016	0	0.0017
2	2	3	-0.0016	0	0.0017	-0.0038	0	0.0008
3	4	5	-0.0038	0	0.0014	-0.0014	0	0.0015
4	5	6	-0.0014	0	0.0015	0	0	0
5	3	7	-0.0038	0	0.0008	-0.0038	-0.0003	-0.0005
6	7	4	-0.0038	-0.0003	-0.0005	-0.0038	0	0.0014

Table 6.5 Forces obtained for one intermediate node in beam and column

Element number	Node 1	Node 2			Forces(KN)			
				Node 1			Node 2	
			x axis(Fx)	y axis(Fy)	Moment	x axis(Fx)	y axis(Fy)	Moment
1	1	2	8.5865	-12.1897	-21.0253	-8.5865	12.1897	-2.7408
2	2	3	8.5865	-12.1897	-2.7408	-8.5865	12.1897	15.5438
3	4	5	-8.5865	-7.8103	-6.8023	8.5865	7.8103	-4.9131
4	5	6	-8.5865	-7.8103	4.9131	8.5865	7.8103	-16.6286
5	3	7	-7.8103	8.5865	15.5438	7.8103	-8.5865	1.6923
6	7	4	-7.8103	8.5865	-1.6923	7.8103	-8.5865	18.8023

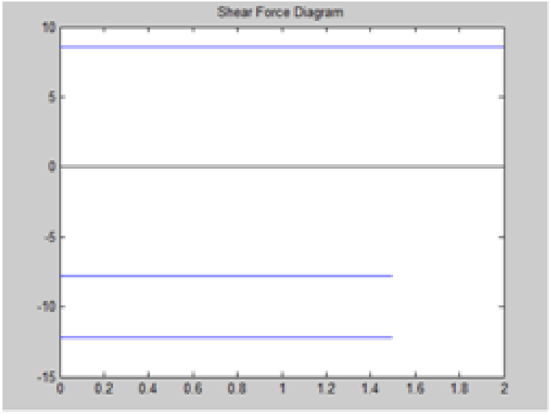

Fig. 6.6 Shear Force Diagram

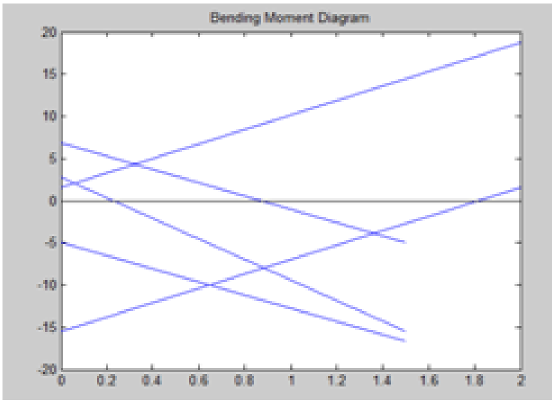

Fig. 6.7 Bending Moment Diagram

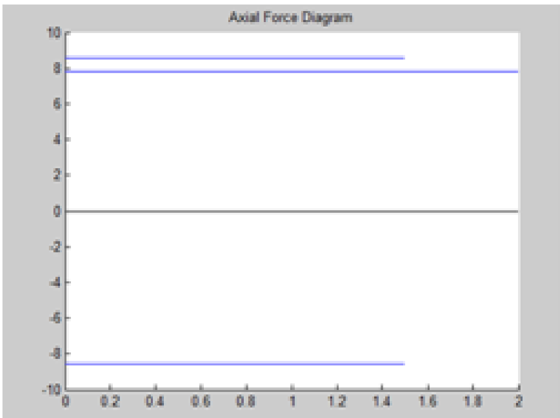

Fig.6.8 Axial Force Diagram

INFERENCE:

Using the MATLAB code, for plane frame in the static analysis we get

- Displacements in x-direction and y-direction
- Rotation
- Forces in x- direction and y-direction
- Moment
- Graph of axial forces, shear forces and bending moment of each element

In case 1 with no intermediate nodes in columns and beams, the deflection at node 2 which is the point of application of force has a deflection of 0.0038 m in negative x direction,0 m in y direction and 0.0014 rotation.

In case 2 when there is one intermediate node in each column and beam, the deflection at node 3 which is the point of application of force has a deflection of 0.0038 m in negative x direction, 0 m in y direction and 0.0014 rotation. The code is verified for the values of deflections. As the system gets on getting complicated, increasing the number of nodes helps in increasing the accuracy and precision of the results obtained.[5]

6.1.2. STUDY OF DEFLECTION FOR DIFFERENT STOREY FRAMES WITH SINGLE BAY

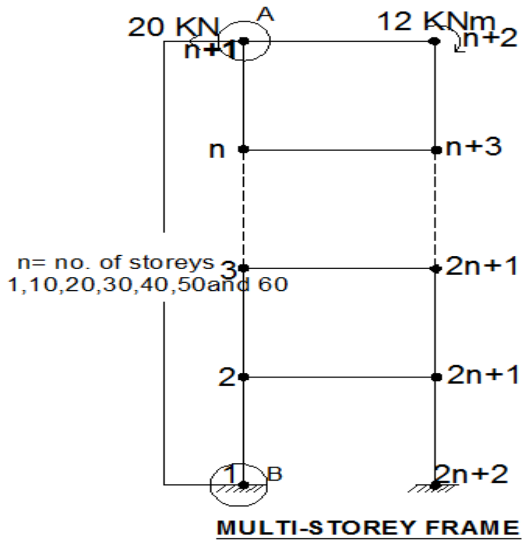

Fig. 6.9 Multi-storey, single bay frame

As shown in the figure, a lateral force given of magnitude 20KN is applied at point A and a moment of 12 KNm at B. We obtain the values of deflection in x- and y-directions and rotations at the point of force application and the values of base moments by changing the number of storeys with number of bays fixed as 1.The results obtained are as follows-

Table 6.6 Deflection results for varying number of storeys

No of Storeys	Deflection in x-axis (m)	Deflection in y-axis (m)	Rotation in z-direction (angle)
1	-0.0038	0	0.0008
10	-0.0791	-0.0006	0.0014
20	-0.1955	-0.0022	0.0022
30	-0.3765	-0.0049	0.0035
40	-0.06543	-0.0086	0.0054
50	-1.0608	-0.0135	0.0078
60	-1.6282	-0.0194	0.0108

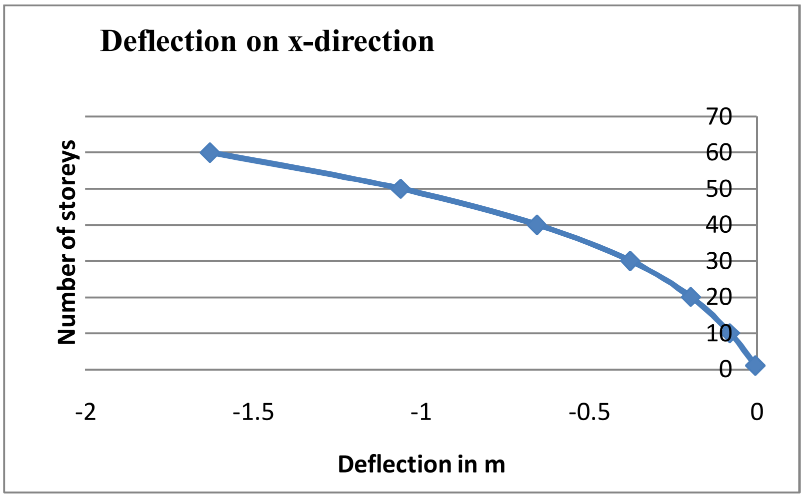

Fig.6.10 Variation of deflection in X-axis

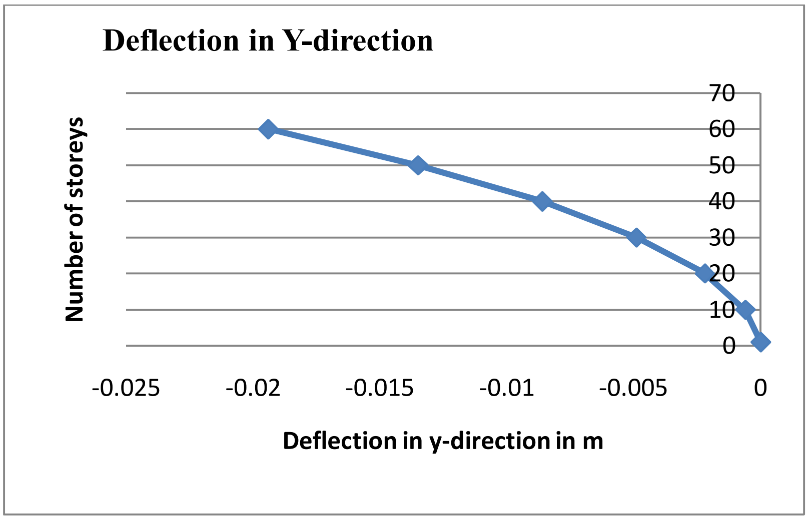

Fig. 6.11 Variation of deflection in Y-axis

INFERENCE:

It is seen that deflection at both x, y axis and rotation increases with increase in number of storeys. This is due to the reason that the stiffness matrix K of the plane frame decreases with increase in number of storeys as K is inversely proportional to length of frame. So as the number of storeys increases, the length of frame increases. As K decreases the displacement matrix 'u' increases which is given by the formula [K]*[u] = [F] where [F] is constant force matrix

6.1.3. VARIATION OF BASE MOMENTS WITH DIFFERENT NUMBER OF STOREYS WITH SINGLE BAY

Table 6.7 Variation of base moments with number of storeys

Number of storeys	1	10	20	30	40	50	60
Moment in Z direction (KNm)	-21.0253	-20.7908	-20.9781	-21.1655	-21.3529	-21.5403	-21.7277

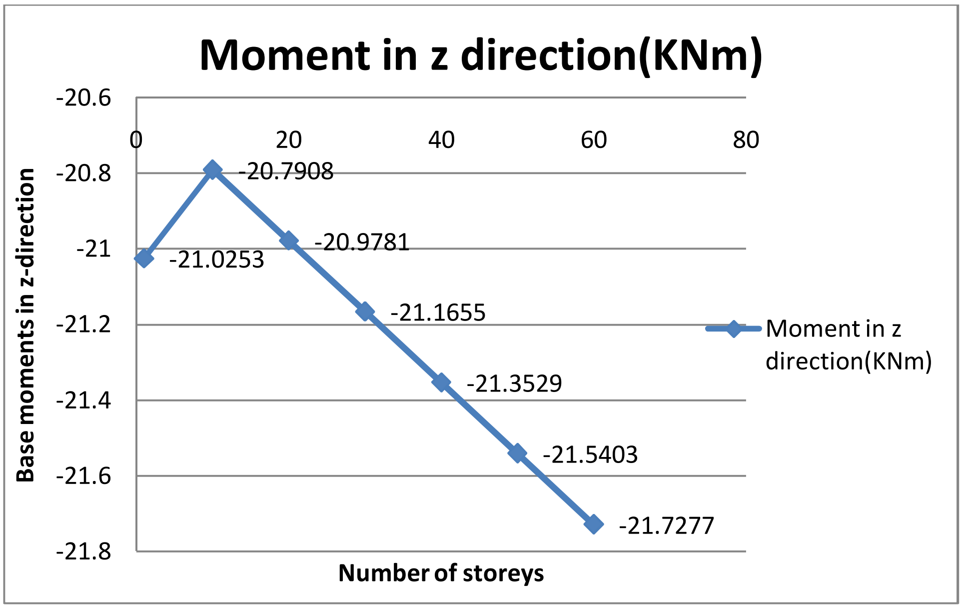

Fig. 6.12 Variation of base moments with number of storeys

INFERENCE

As the number of storeys increases, the negative base moment increases. This is due to increase in length of frame. It is so because as the length of the frame increases, the moment arm of the force increases and so does the base moment increases.

6.1.4. VARIATION OF BASE MOMENTS WITH DIFFERENT NUMBER OF BAYS AND FIXED NUMBER OF STOREYS

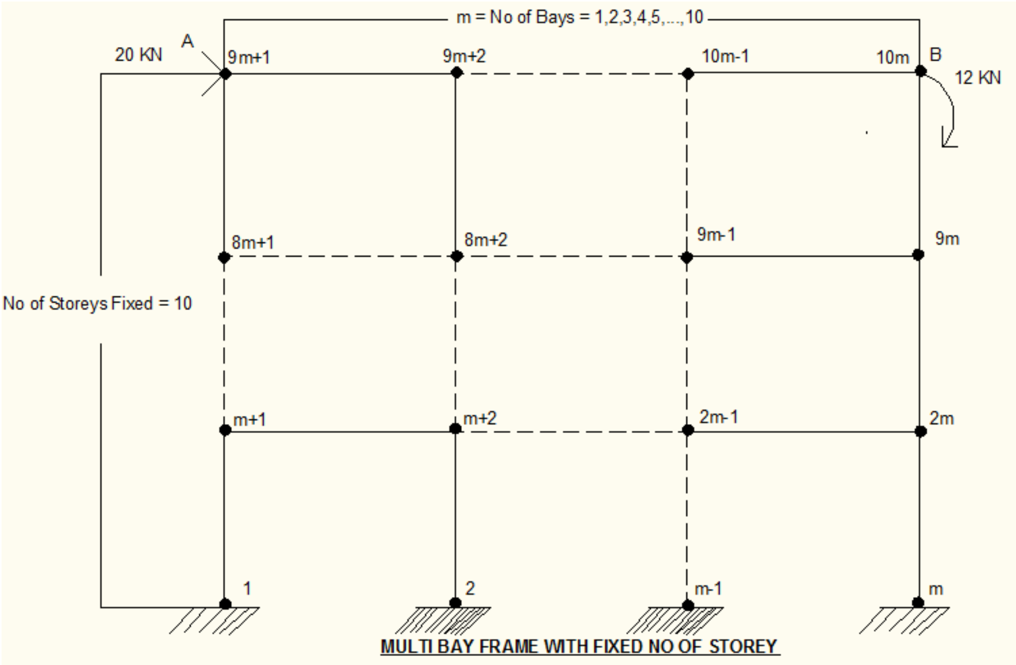

Fig. 6.13 Multi-bay frame with 10 storeys

The number of storeys is fixed as 10.

Table 6.8 Variation of base moments with number of bays

Number of bays	1	2	3	4	5	10
Moment in Z direction (KNm)	-20.7908	-12.3950	-8.8966	-6.9372	-5.683	-1.9231

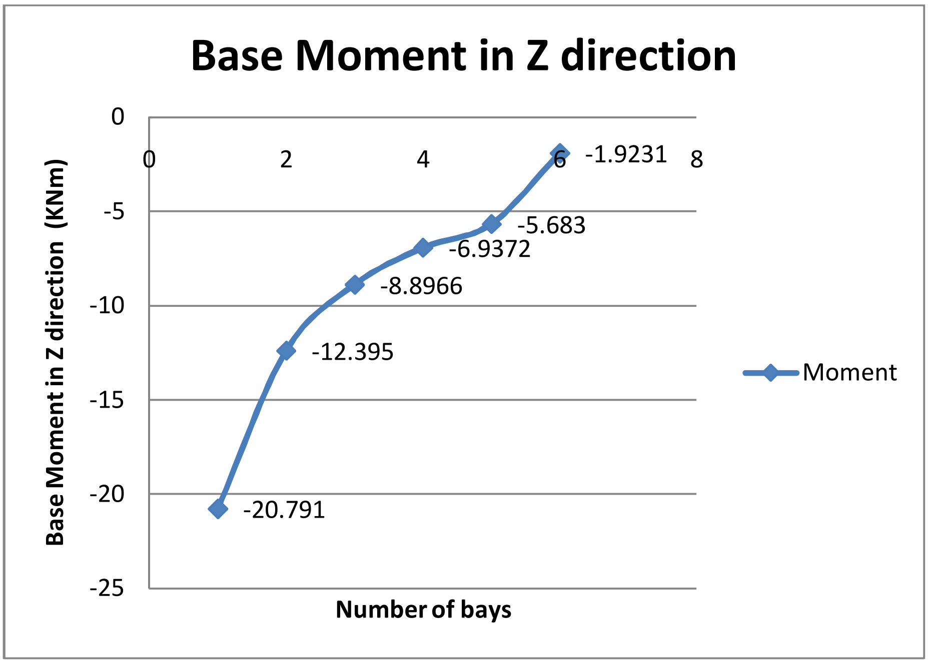

Fig. 6.14 Variation of base moments with number of bays

INFERENCE

As the number of bays increases, the negative base moment decreases. This is due to increase in area of the frame. As the area increases, the base moment is distributed over a larger area and hence the decreasing pattern.

6.2. DYNAMIC ANALYSIS

The MATLAB code is run for different storeys and different bays buildings for dynamic analysis of the frame structure. The free vibrational frequency is calculated at each node and the forced vibration analysis is done under different load condition.

6.2.1. COMPARISON OF EXPERIMENTAL AND NUMERICAL ANALYSIS RESULTS

The two dimensional dynamic analysis done using MATLAB program for plane frame element is verified by comparing the numerical results obtained with the experimental results obtained using FFT analyzer in the structural laboratory. The experiment was done on a single storey and single bay frame made up **aluminium** using FFT analyser and PULSE software.

NUMERICAL ANALYSIS OUTPUT:

Table 6.9. Details of the experimental model

Modulus of Elasticity, E	70 GPa (Aluminium)
Density ρ	2700 kg/ m^3
Cross section of element b*d	.025x0.035m^2
Moment of Inertia, I	$9 * 10^{(-7)}$ m^4
Area of cross-section, A	$8.75 * 10^{(-4)}$ m^2
Length of the column	.45 m
Length of the beam	.30

OUTPUT

Total no of intermediate nodes in column=0
Total no of intermediate nodes in beam=0
total no of nodes in each column is nct = 2

total no of nodes in frame element is n = 4.

total no of elements in the frame is e = 3.

freq = 4.7603 27.6349 55.9106 75.2554 82.7566 196.4153

EXPERIMENTAL ANALYSIS OUTPUT

Natural frequencies obtained are: 8 32 52 68 76 196

TABULATION

The natural frequencies obtained through numerical and experimental analyses are tabulated as follows:

Table 6.10 Comparison of numerical and experimental analysis results

NUMERICAL ANALYSIS	EXPERIMENTAL ANALYSIS
4.7603	8
27.6349	32
55.9106	52
75.2554	68
82.7566	76
196.4153	196

INFERENCE

The results obtained for natural frequencies for the specified frame through experimental analysis and numerical analysis are in coherence with each other. Thus the numerical procedure is validated.

6.2.2. STUDY OF VARIATION OF NATURAL FREQUENCY WITH CHANGING NUMBER OF STOREYS AND FIXED NUMBER OF BAY

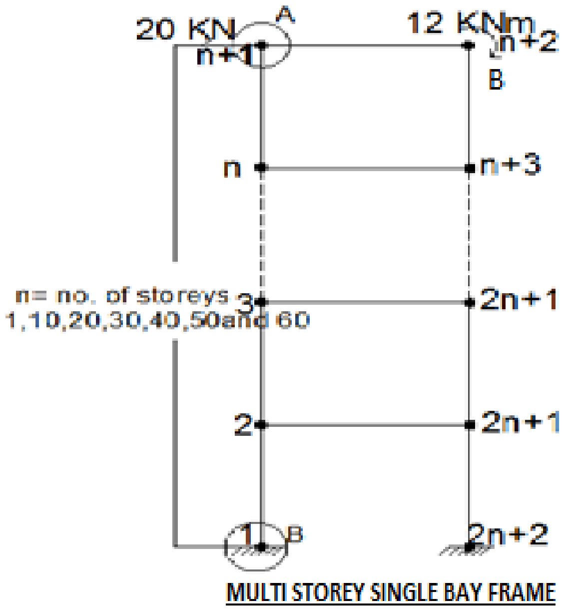

Fig. 6.15 Multi-storey, single bay frame

The results obtained are as follows-

Table 6.11 Variation of Modal Frequencies with number of storeys

Number of storeys	Mode 1	Mode 2	Mode 3	Mode 4
1	2.5656	9.5564	22.789	69.8116
10	0.2025	0.6243	1.091	1.6279
20	0.0942	0.2891	0.5102	0.7308
30	0.0577	0.1811	0.3398	0.4708
40	0.0393	0.1282	0.2413	0.346
50	0.0284	0.0973	0.1884	0.2723
60	0.0214	0.0772	0.1532	0.225

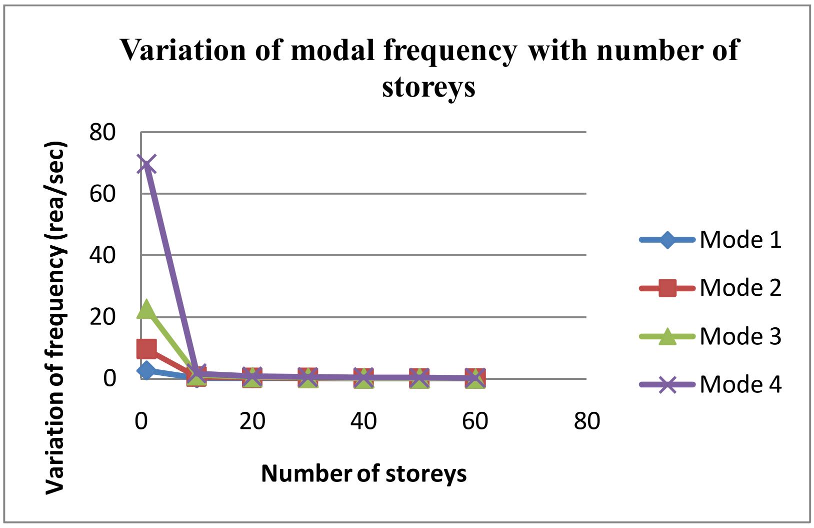

Fig. 6.16 Variation of modal frequency with number of storeys considering 1-60 storeys

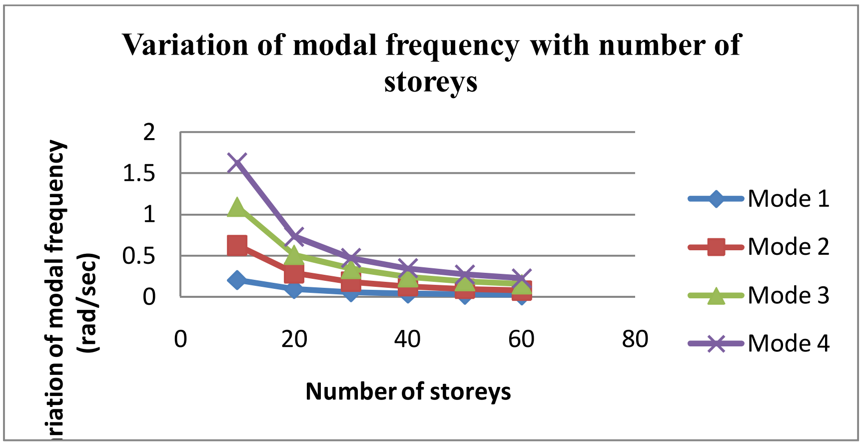

Fig.6.17 Variation of modal frequency with number of storeys considering 10-60 storeys

INFERENCE

As the number of storeys increases, the length of the frame element increases and hence stiffness [K] decreases as it is inversely proportional to length. As [K] decreases, the angular frequency ω decreases as it given by relation [K] – ω2[m]=0.

Hence for short building the angular frequency is very high an so it does not fall down during severe earthquake. It is since the frequency of the earthquake doesn't match with natural frequency of building. In case tall buildings, the natural frequency is low ; hence easy collapse under earthquake.

6.2.3. STUDY OF VARIATION OF NATURAL FREQUENCY WITH CHANGING NUMBER OF BAYS AND FIXED NUMBER OF STOREYS

Number of storeys is kept fixed as 10**.**

Table 6.12 Variation of Modal Frequencies with number of bays

Variation of Modal Frequencies with number of bays

Number of bays	Mode 1	Mode 2	Mode 3
1	0.2025	0.6234	1.091
2	0.2069	0.6331	1.0973
3	0.2089	0.6377	1.0995
4	0.21	0.64	1.1004
5	0.2106	0.6415	1.101
6	0.212	0.6446	1.1022

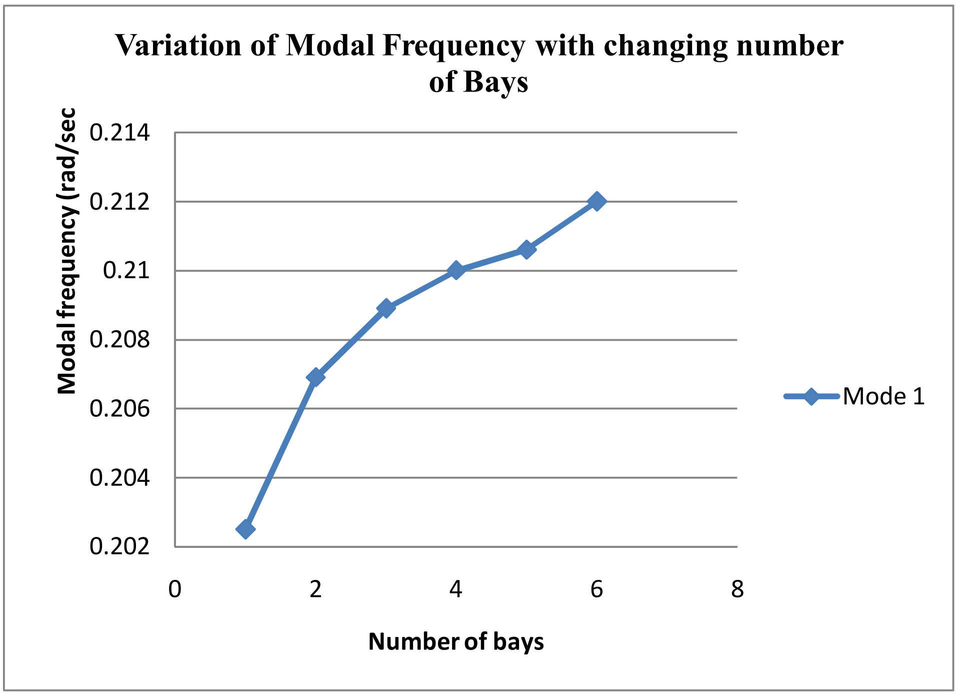

Fig. 6.18(a) Variation of Modal Frequency with changing number of bays: Mode 1

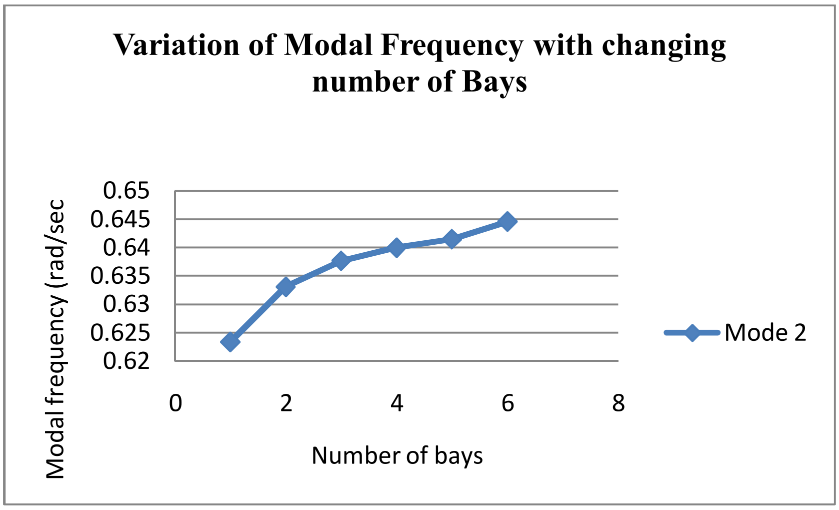

Fig. 6.18(b) Variation of Modal Frequency with changing nu8mber of bays: Mode 2

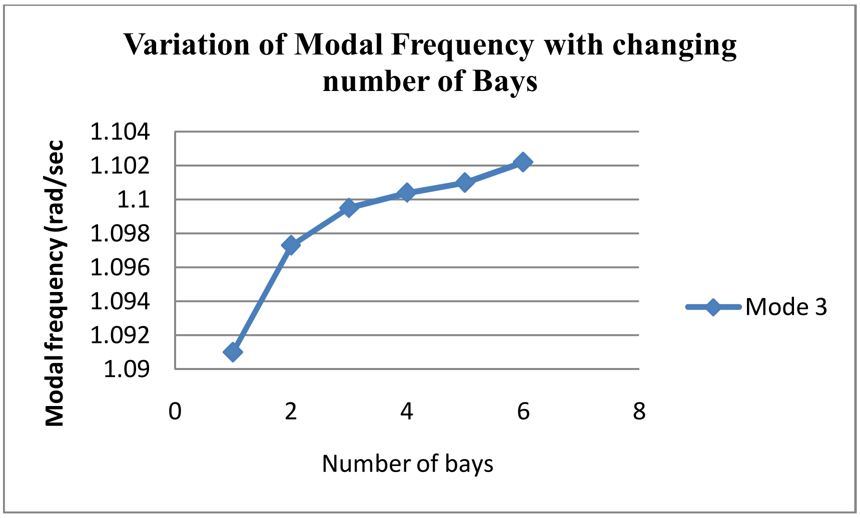

Fig. 6.18(c) Variation of Modal Frequency with changing number of bays: Mode 3

INFERENCE

As the number of bay increases, the area of frame element increases. Hence the stiffness [K] increases as it is directly proportional to area. As [K] increases, the angular frequency ω increases as it given by relation [K] – ω2 [m] =0.

Hence by increasing the number of bays ,an increasing pattern in modal frequencies is observed.

6.2.4. DYNAMICS ANALYSIS UNDER DIFFERENT LOAD CONDITIONS

6.2.4.1. SYSTEMS WITHOUT DAMPING

CASE 1: Arbitrary Load on Ground

For the first problem, we consider a twenty-storey building with a single bay and the load considered on the ground is **Elcentro data and IS Code Data** which has different forces acting at different time.

Table 6.13 Assumed data for the system

Modulus of Elasticity, E	210 GPa (Steel)
Moment of Inertia, I	$5 * 10^{(-5)}$ m^4
Area of cross-section, A	$2 * 10^{(-2)}$ m^2
Length of the column	3 m
Length of the beam	4 m
Density, ρ	7850 kg/ m^3

(a) <u>ELCENTRO DATA</u>

Elcentro is metropolitan city of America. The ***Elcentro data*** is the data of the earthquake which occurred at Elcentro during May 18, 1940. The data consists of two column acceleration and time.

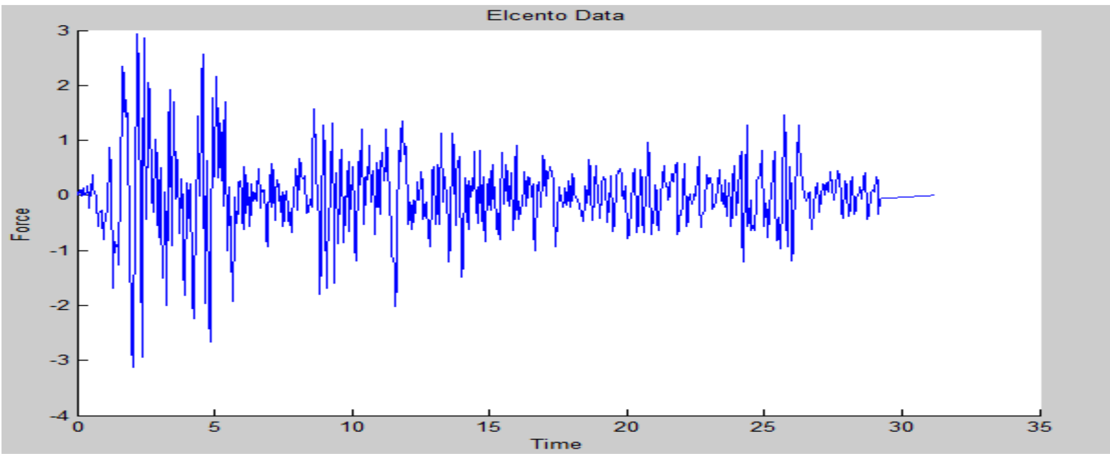

Fig. 6.19 Elcentro force data

OUTPUT

Total number of intermediate nodes in each column= 1

Total number of intermediate nodes in each beam = 1

Number of storeys= 20

Number of bays = 1

Total number of nodes in the plane frame=102

Total number of elements in the plane frame=120

The mass matrix, stiffness matrix and free vibration frequency is too large to be shown.

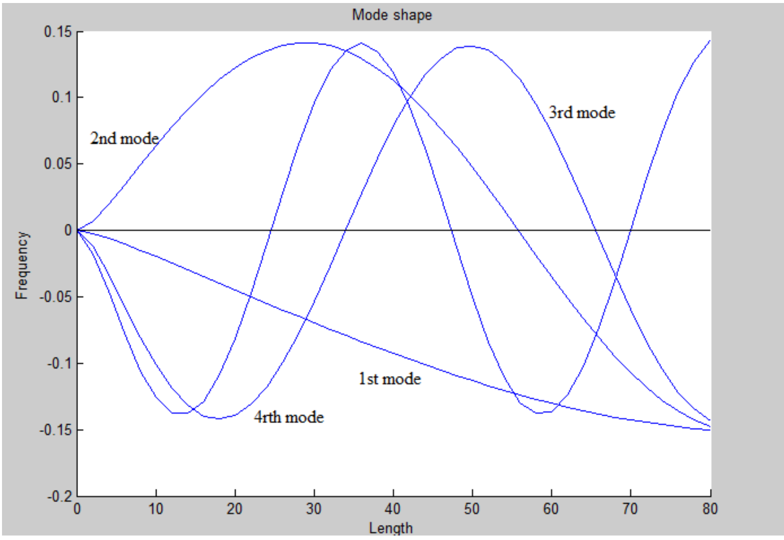

Fig.6.20 Mode diagram for the system

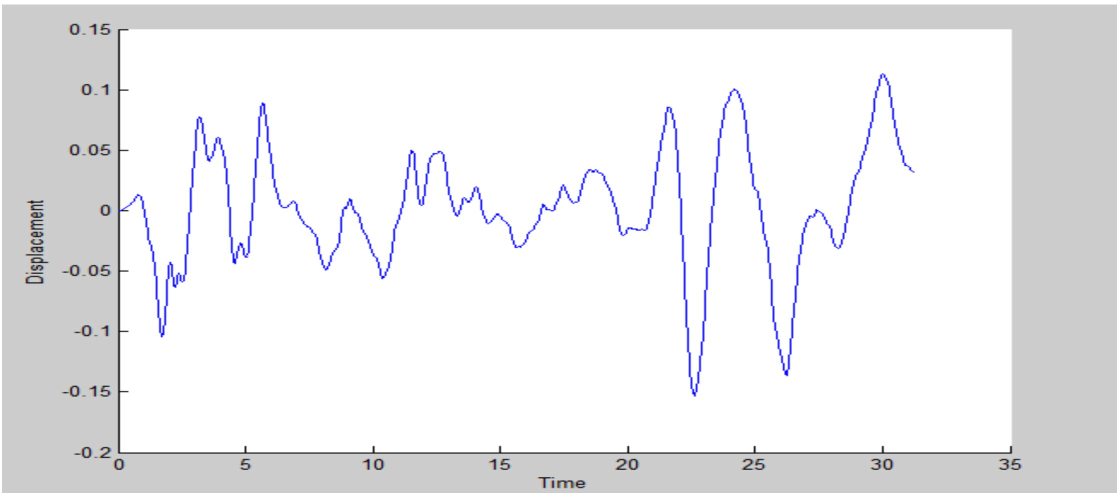

Fig. 6.21 Forced Vibration Deflections Due To Arbitrary Load on Ground (Elcentro force data): Undamped System

(b) IS CODE DATA

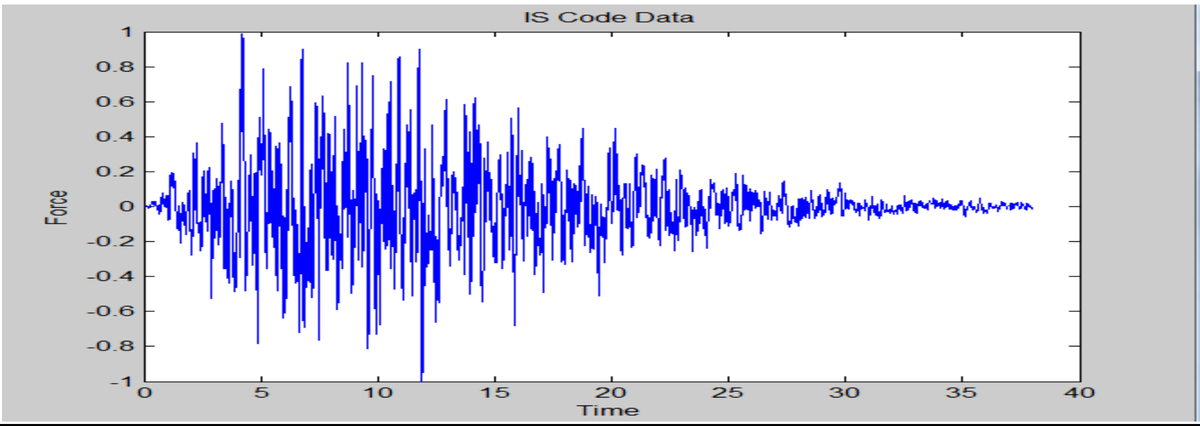

Fig.6.22 IS Code Force Data

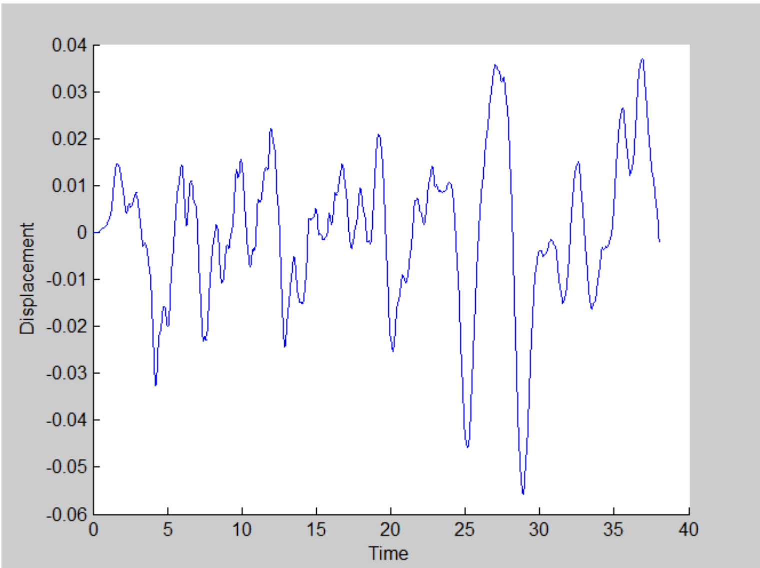

Fig.6.23 Forced Vibration Deflection Due To Arbitrary Load On Ground (IS Code): Undamped System

RESULT:

The maximum earthquake force acting in the IS Code data is 1N and the maximum displacement we obtain is 0.06 m. While in the case of Elcentro data where the maximum force is 3N, the maximum displacement comes out to be 0.15m.

CASE 2: Load on the frame

Here we consider a twenty-storey building with a single bay but here the load considered is applied on the frame. A force of 20 KN is considered on top left node and a moment of 12 KNm on top right node of frame.

OUTPUT

Total number of intermediate nodes in each column= 1

Total number of intermediate nodes in each beam = 1

Number of storeys= 20

Number of bays = 1

Total number of nodes in the plane frame=102

Total number of elements in the plane frame=120

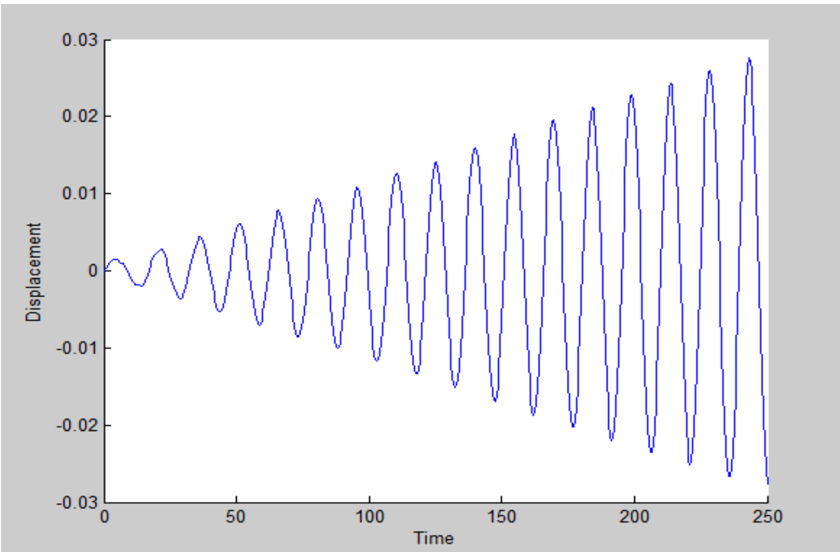

Fig.6.24 Forced Vibration Deflection Due To Load on Frame: Undamped System

CASE 3: Sinusoidal load on ground

Here we consider a twenty-storey building with a single bay but the load considered is a sine function of angular frequency. Hence the load taken is F= A*sin(ωt) whereA= amplitude.; ω= free vibration frequency of frame element.; t = time period

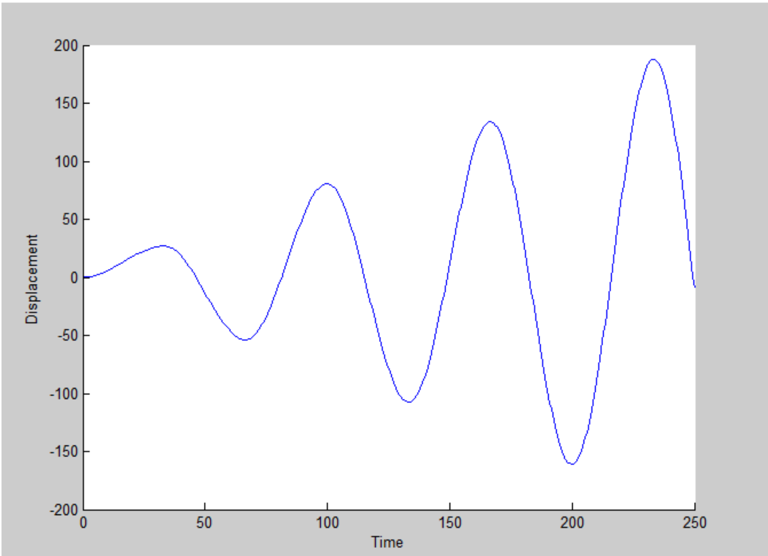

Fig.6.25 Forced Vibration Deflection Due To Sinusoidal Load on Ground: Undamped System

INFERENCE:

Using the program in MATLAB code the above output is generated. . The various mode shapes of fundamental frequency for frame element are obtained using free vibration frequency which remains same under various load conditions as the Eigen values derived are functions of mass and stiffness matrix. Under forced vibration condition the nature of graph changes as the nature of load changes. A resonance graph is obtained when the forced vibration frequency matches with natural vibration frequency of frame element at which we get maximum displacement.

6.2.4.2. SYSTEMS WITH DAMPING

In damped condition, a damping factor C is introduced in forced vibration analysis of same multi-storey and multi bay frame. In this section for different conditions of load, we have represented only the deflection graphs because we obtain the same mode shape diagrams as in the previous case.

CASE 1: Arbitrary Load on Ground

(a) <u>ELCENTRO DATA</u>

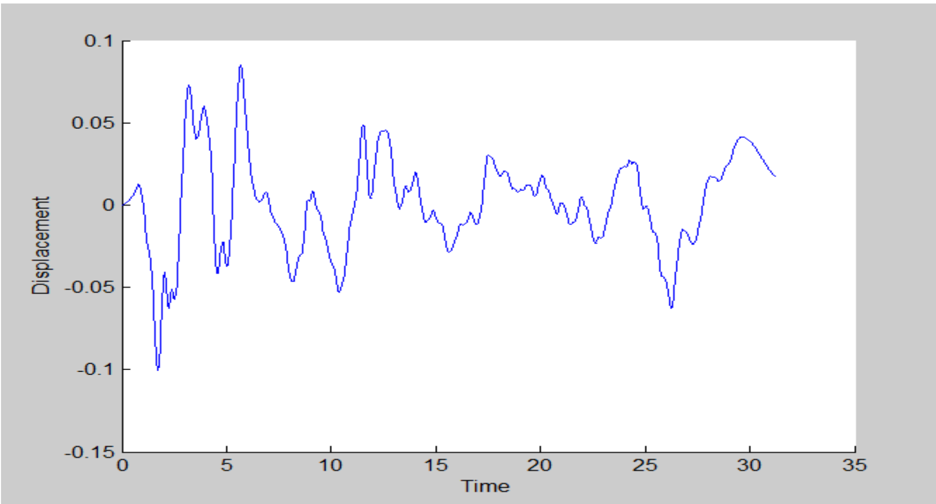

Fig. 6.26 Forced Vibration Deflections Due To Arbitrary Load on Ground (Elcentro force data):Damped System

(b) <u>IS CODE DATA</u>

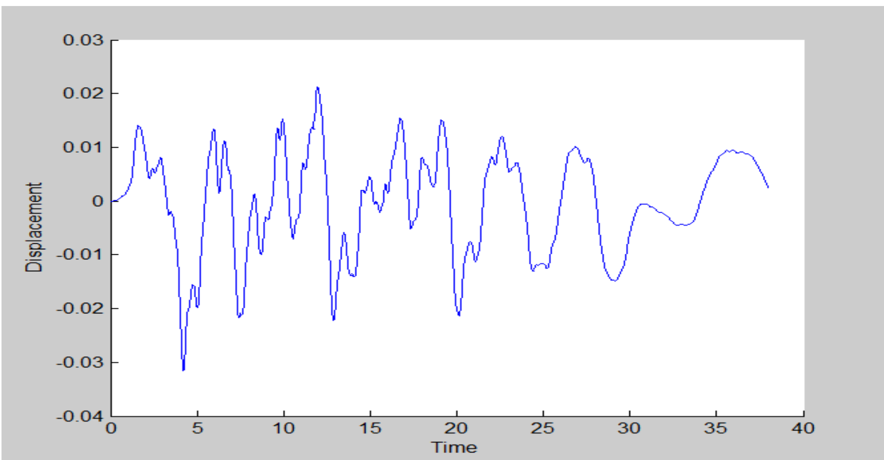

Fig 6.27 Forced Vibration Deflection Due To Arbitrary Load On Ground (IS Code): Damped System

CASE 2: Load on the frame

A force of 20 KN is considered on top left node and a moment of 12 KNm on top right node of a twenty-storey building with a single bay.

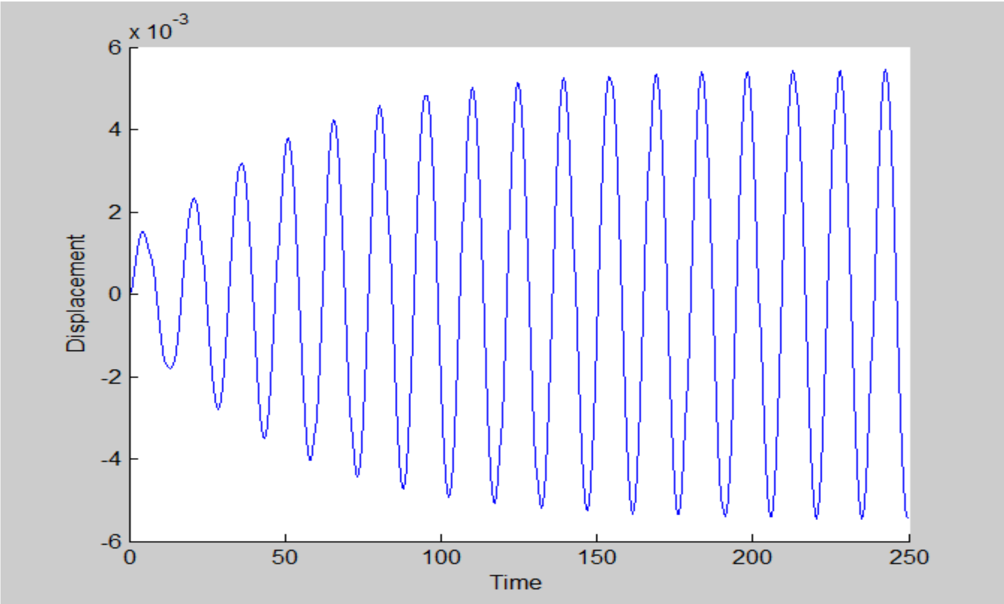

Fig. 6.28 Forced Vibration Deflection Due To Load on Frame :Damped System

CASE 3: Sinusoidal load on ground

F= A*sin(ωt)

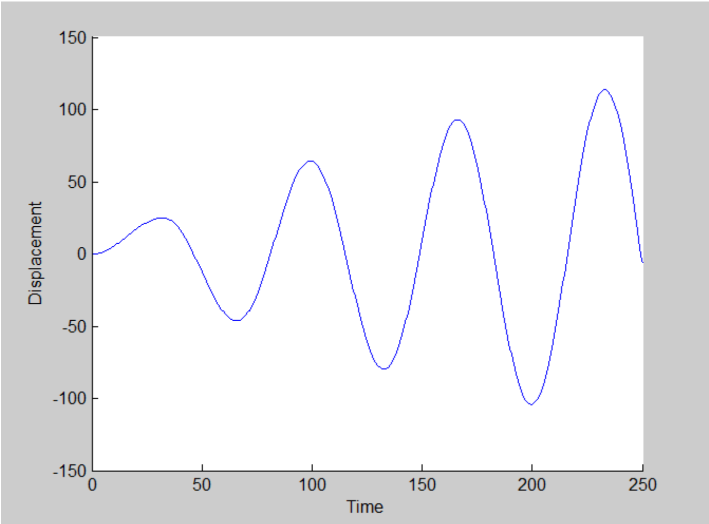

Fig. 6.29 Forced Vibration Deflections Due To Sinusoidal Load on Ground: Damped System

INFERENCE:

The maximum deflections obtained in each case of loading for damped and undamped conditions have been tabulated as follows-

Table 6.14. Comparison of results in undamped and damped conditions

	UNDAMPED CONDITION	DAMPED CONDITION
ARBITRARY LOAD USING ELCENTRO DATA	0.15	0.1
ARBITRARY LOAD USING IS CODE DATA	0.06	0.03
LOAD ON FRAME	0.03	0.006
SINUSOIDAL LOAD ON THE GROUND	.002	.0012

From the various displacement graphs obtained in case of both damped and undamped conditions, it is seen that we get lesser values of deflection in damped conditions. Hence dampers when used in a multi-storey and multi-bay building reduce displacement under different loading conditions.

7. CONCLUSION:

From the above study and results it has been found that finite element method is very useful in analysis of frame structures. A code was developed in matlab for analysis of two dimension frames under arbitrary loading. The result obtained are the dynamic and static properties two dimensional frame structures such as natural frequency of the structure, shear force , bending moment at any storey of frame structure. It has been seen that when the forced vibrational frequency matches with natural frequency of the structure resonance peak occurs. The result obtained from the code can be used for designing of multi-storey and multi-bay frame. The variation in code generated result and experimentally calculated result is very less which validates the used code.

8.REFERENCES:

[1] Ali Ugur Ozturk and Hikmet H. Catal, "*Dynamic Analysis of Semi-Rigid Frames"*, Mathematical and Computational Applications, 10, No. 1,(2005.) pp. 1-8,

[2] http://www.me.berkeley.edu/~lwlin/me128/FEMNotes.pdf , "Introduction to finite element modelling,"

[3] Miodrag Sekulovic, Ratko Salatic, Marija Nefovska, "*Dynamic analysis of steel frames with flexible connections*", Computers and Structures 80 (2002): pp 935-955, www.elsevier.com

[4] Shosuke Morino and Yasuhiro Uchida, "*Dynamic Response of Steel Space Frames Under Earthquake Excitation in Horizontal Arbitrary Direction*"

[5]Kattan P.I., "MATLAB Guide to Finite Elements-An interactive approach", online library, http://www.springer.de/engine/

[6] Chandrupatla, T.R.; Belegundu,A.D, Introduction to Finite Elements in Engineering, Prentice Hall of India Private Limited

[7] Paz,M., "Structural Dynamics

[8] Kurt Mcmullin,http://www.engr.sjsu.edu/mcmullin/courses/ce265/tutorials/mode.htm;

[9] Chopra,Anil.K. , Dynamics of Structures-Theory and Applications to Earthquake Engineering, Prentice Hall of India Private Limited, New Delhi.

[10] http://www.scribd.com/doc/39994829/Dynamics-Analysis-by-Numercial-Integration; Dynamic Analysis by Numerical Integration.

[11]http://nptel.iitm.ac.in/courses/Webcoursecontents/IIT%20Kharagpur/Structural%20Analysis/pdf/m4130.pdf , 'Direct Stiffness Method: Plane Frames'